Tutorial

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ENC 4265

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Introduction

The need for engineers is always increasing in this day and age. With technology and more advanced resources, engineers are expected to learn how to use new software constantly to help make their jobs easier and more accurate. SolidWorks is one of the most popular software in the engineering industry and is always being updated for engineers to use in the workplace. The subject of the tutorial will be all things SolidWorks including how To's for building specific parts utilizing the different features, a refresher incorporating AutoCAD (a sister 2D software), and potential advance uses. This tutorial will be catered towards current or prospective engineers, including high school and university students. The terminology used will also be in accordance with basic understanding of math principles used in engineering such as algebra, geometry, trigonometry, and calculus. Learning and potentially getting certified in SolidWorks will help adults in the future get jobs, as well as earn higher pay because of their higher qualifications compared to their coworkers.

More Information

For more information on SolidWorks, visit https://www.solidworks.com/.

Materials

This tutorial will be based on SolidWorks 2019, but can comply with updated versions as well. In order to get SolidWorks, visit https://www.solidworks.com/, to pay for and download the software. If you are with a school or business, you can redeem a code provided by your organization to get a different kind of SolidWorks service. It is recommended to give yourself a couple hours to let the technical requirements be checked on your device, prior to dashboard set up. SolidWorks should already be installed and logged in before proceeding with this tutorial.

Purpose

The purpose of this tutorial is for educational purposes only. It is not meant to endorse the product but serve as a guide for experienced engineering personnel. For those looking to add more experience for their personal and professional development, this tutorial will help the future use of this software in your designated field of the different engineering disciplines.

2D VS. 3D

Engineers use both 2D and 3D modeling software all the time, but some are more suitable for different projects in the field. SolidWorks gives the user the ability to perform both within one application. No separate documentation necessary, all measurements and scaling can now be done within a single file.

2D Design

Software, such as AutoCAD, is most popularly found in the form of blueprints. These are 2D sketches that give a layout for a three-dimensional build yet to come. The importance of these softwares, is that it checks the accuracy of the dimensions before proceeding in a construction process where something is more likely to go wrong with the increase of materials. Using sketching software allows professionals to make fewer mistakes and have more exact measurements before starting something with more permanent consequences. For example, for construction workers making a new building that is more than just the standard four wall design, it is important to test these dimensions. If the numbers that are being produced don't correspond to each other correctly, then it is time to make some changes to the plan.

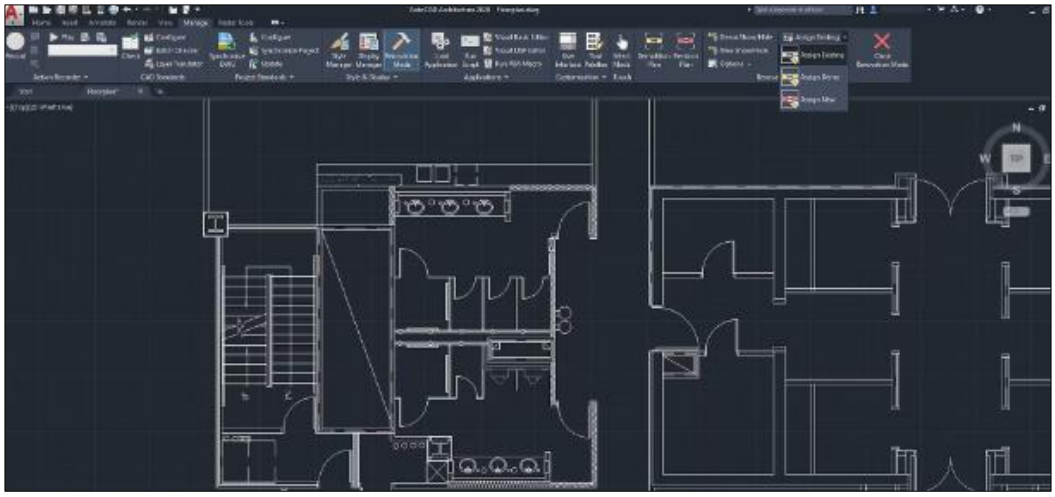


Image by AutoDesk.

3D Design

New software, like the always updating SolidWorks, takes 2D sketching even further. While the sketches are important, for production features it is important to know how other aspects of a project will work. These things could include parts that are being made that ae going to directly interact with other parts.

SolidWorks allows for both parts to be designed and their 3D dimensions to be measured and tested while working with one another. This allows for engineers to fix any mistakes that could have happened in the future, such as not fitting, too big or small, or areas that get stuck and are not functional. All of these test factors that would have traditionally been discovered after prototypes are made would have made the engineering process longer with having t redevelop and design parts that don't have these problems. 3D software makes the engineering process faster and quicker.

Image by The SolidWorks Blog.

Certification

SolidWorks offers engineers the opportunity to get certified in this software. There are different ways to get tested and various levels of certification. Earning one of these will show employers that you are knowledgeable in this area and have credibility. This can also help lead to pay raises within a company because of the demonstration of furthering education and skills. This will make you more valuable compared to employees that don't have the certification. If there is ever an opportunity where you can get certified in something, earn that certification. This will help you a lot in the workplace in many areas. Some websites allow you to take practice exams in preparation for your certification which is helpful, since the SolidWorks certification tests each follow certain types of format that are not in traditional testing. It is important to get familiar with the testing layout beforehand so you can maximize your chances of passing, and not wasting time figuring out the test itself.

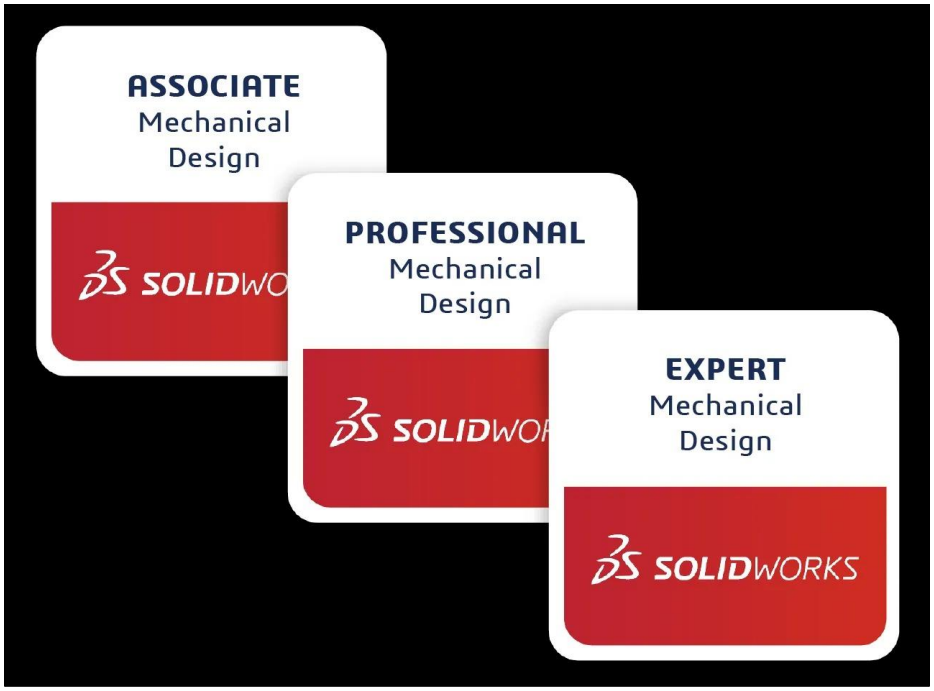


Image by GoEngineer.

File Types

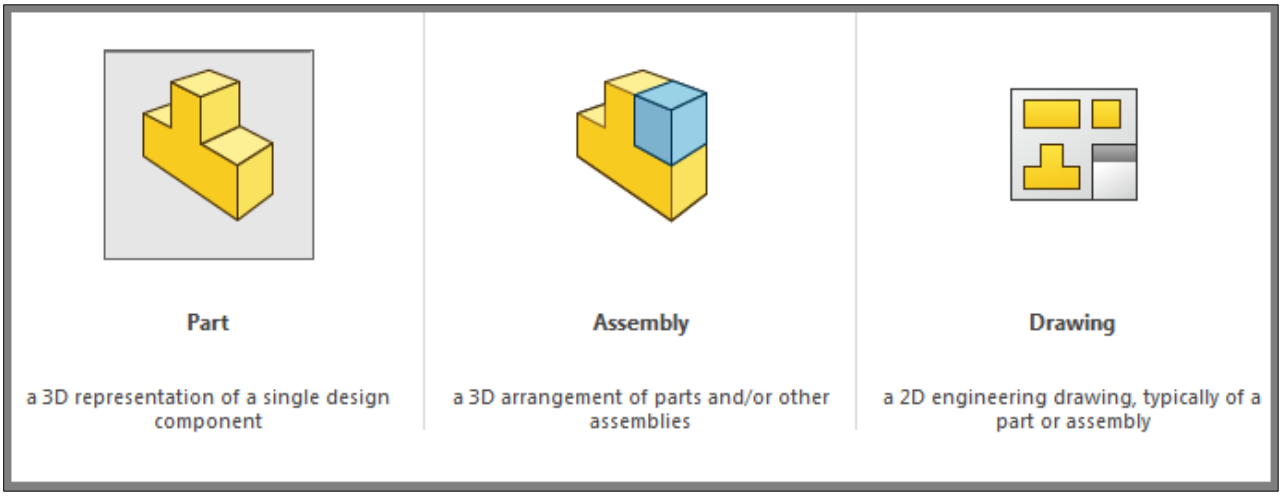


Image by Angelle Erickson.

Part

This is the most common file type. Since SolidWorks is a 3D based modeling type, engineers need to make every single part before putting it together. This helps engineers track any necessary changes that may need to be made to an individual part, after seeing it work with others. This saves time so engineers don't have to remake everything all over again in the testing process.

Assembly

After engineers design all their parts, it is time to open an assembly file where they can insert the part files one by one. Here you can simulate functionality by measuring movement and limitation. There are special commands that are within the Assembly file types that allow engineers to make defined specifications that will demonstrate how the part will move in the real world.

Drawing

Much like AutoCAD, this is a special file type that is 2D drawings that are best used for the schematics of future parts. While you can sketch in the other file types, engineers find having separate blueprints important for their records. If anything does not go as planned, engineers can trace back to their sketches to find the root of the problem.

Features in the Files

With each different type of file, the Command Bar changes. What appears in a Part file may not be in an Assembly file, so it is important to make the appropriate changes all in one because

that option may not be available when working later. It is always best to familiarize yourself with the different features before beginning a new project.

Color

When designing and building different parts, it can help to color code them. The default color on part files is a shade of grey and this can make it difficult to decipher which part is what. SolidWorks gives engineers the option to change the color of each part and even the background so it can make the designs easier to track and more visually appealing.

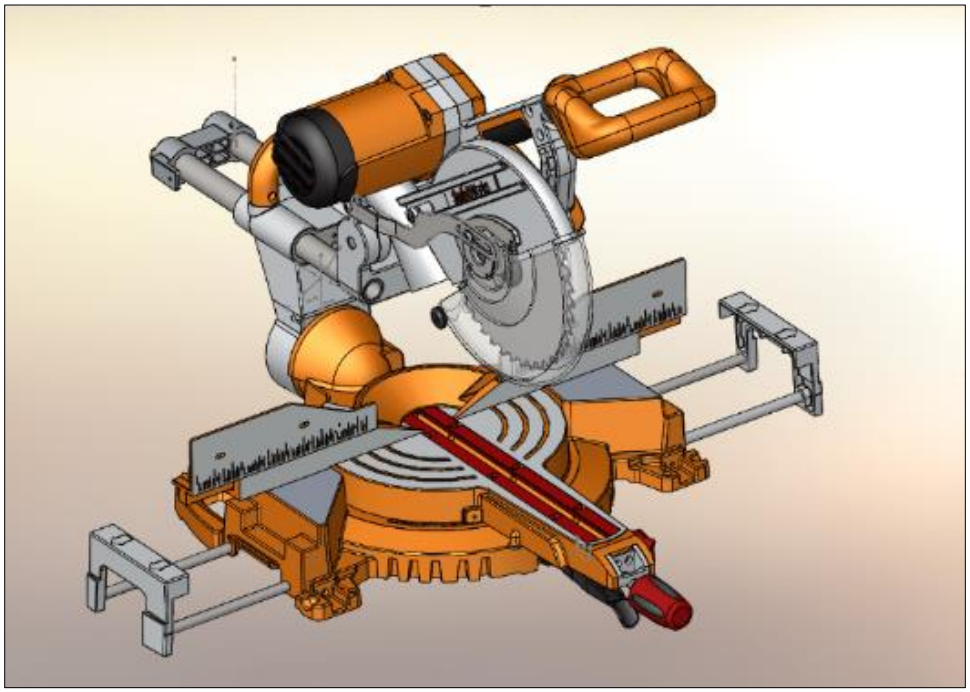


Image by Hawk Ridge Systems.

Part Materials

Similar to the colors feature, SolidWorks allows you to apply different materials to the parts as well. For example, you can apply different kinds of wood, plastic, and metals to make the 3D model look more lifelike. Engineers can find this feature beneficial when they must present their files to others and adding materials can give their audience a better picture of what the final product will look like.

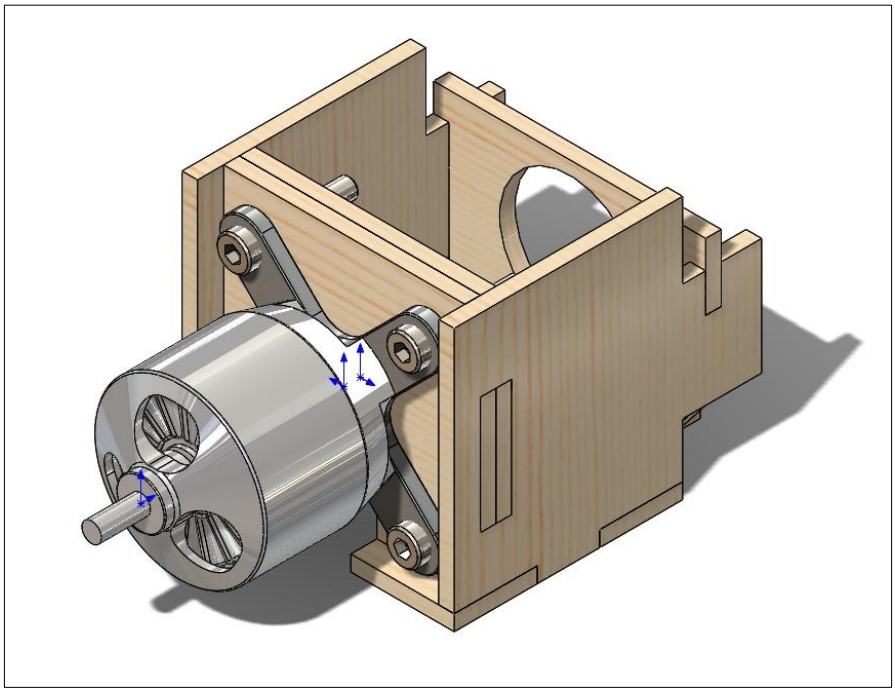


Image by SolidWorks Blog.

Graphics Area

This is the large open space that the parts and assemblies appear. It takes up most of the screen and can be altered by special features.

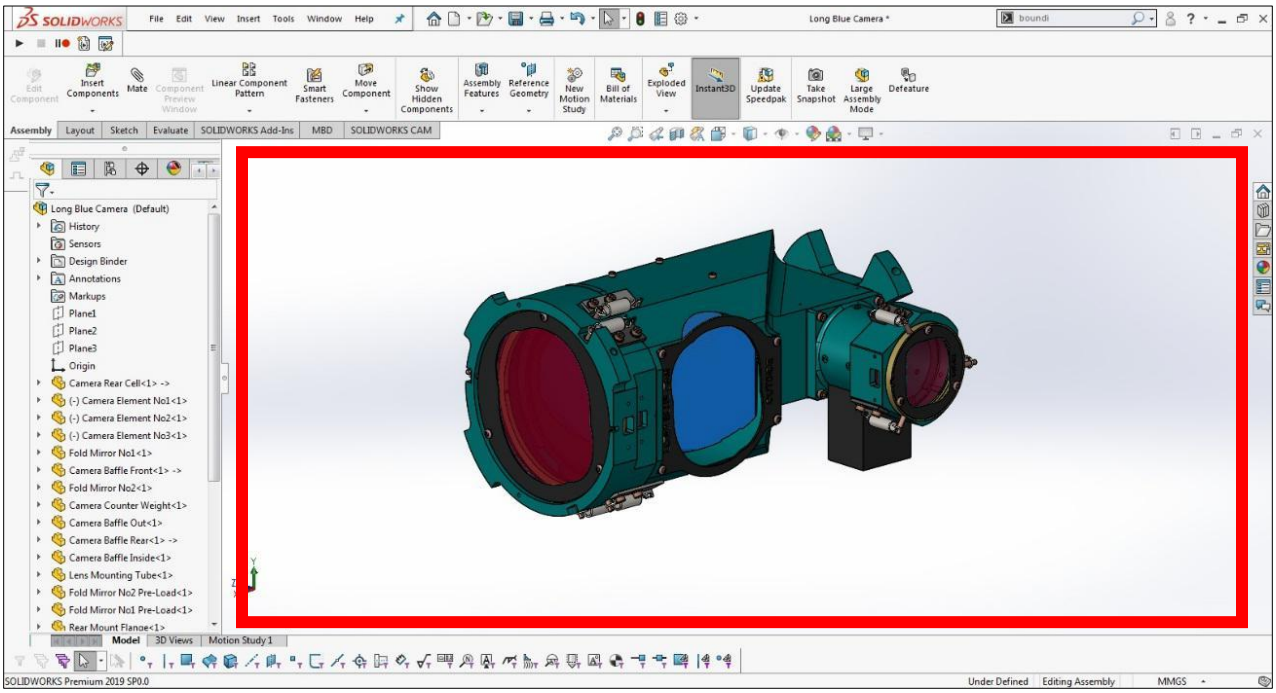
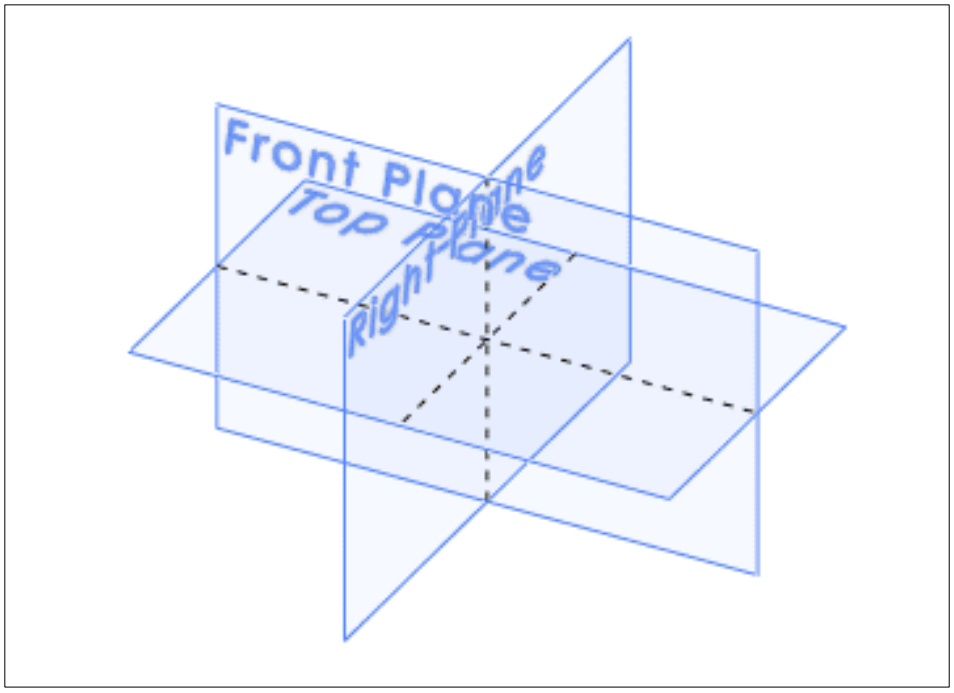


Image by SolidWorks Blog.

Planes

There are three planes that are automatically shown when opening any type of file: Front, Side, and Top. Picking the plane is the first step of designing since it affects commands that are often utilized later, such as rotating or viewing of plane.



Feature Manager Design Tree

This is on the left side of the screen going vertical to show the order of parts, mates, and other details. SolidWorks implementation of this feature is helpful because if there are complications that the software catches, it can highlight which aspect of the file is causing the problem. This saves time by canceling out the investigation, and allows for changes to be made without affecting the rest of the file. It is always used and makes organizing much easier.

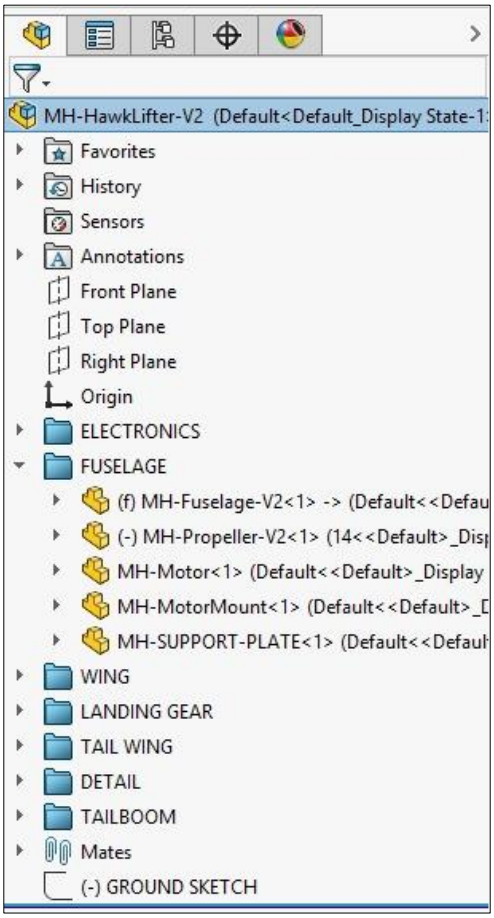


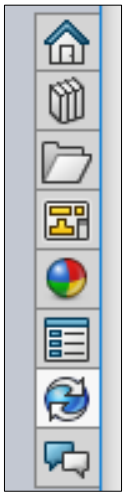
Image by Hawk Ridge Systems.

Standard Toolbar

The save button will be your must used button in the entire program. It is important to get into the habit of saving your work as you go along and not just when you are finished. You never know when the technology you are using will be affected by a power outage, slow internet, etc. It is also important to get familiar with the rest of the buttons in the toolbar. This is located at the top middle area of the entire screen, not the graphic area where the part appears.

Task Plane

This small bar can appear on the top of the graphics area, where the part appears, or on the left hand side. It is a smaller bar that includes features that will not be primarily in use for the designing, but can be helpful with its specific features. For example, the appearances icons appears here. It is not a vital feature, but it can be a preference when putting the finishing touches on a file.



Creating a Square Catapult Base

Making the Parts

1. **Open desktop and click on SOLIDWORKS icon.**

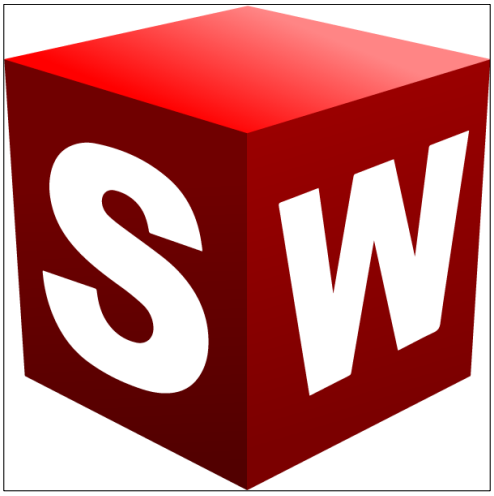


Image by CAD Dimensions.

2. Once SOLIDWORKS finishes loading three options will appear: Part, Assembly, and Drawing. **Click on "Part" and then "OK."**

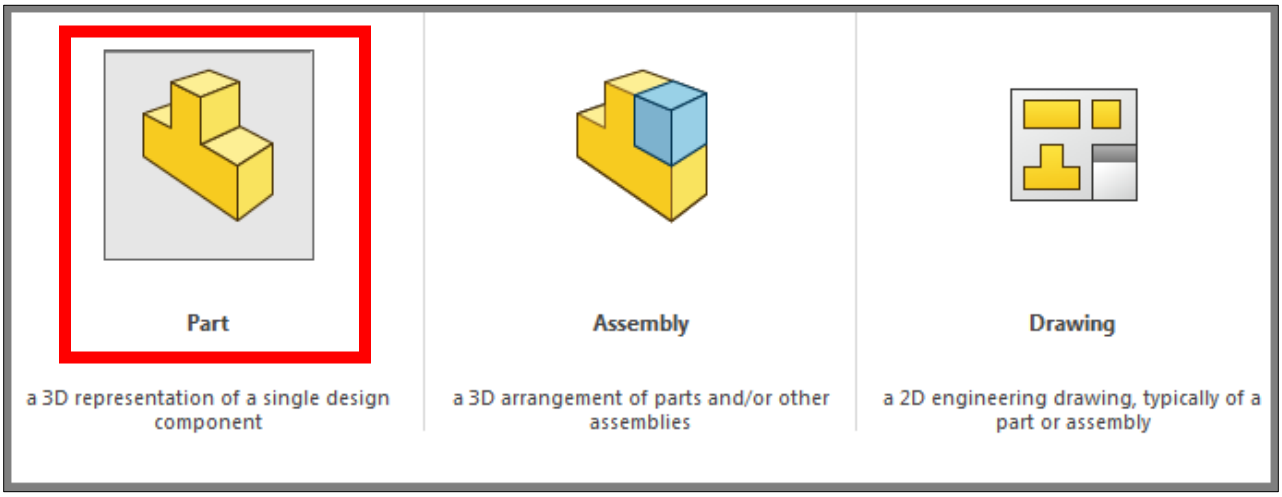


Image by Angelle Erickson.

3. Screen should automatically load blank space with toolbars on top and left side of screen. On top lefthand corner of screen, a small tab "Sketch" will appear next to "Features." **Click "Sketch" tab, then first icon "Sketch."**

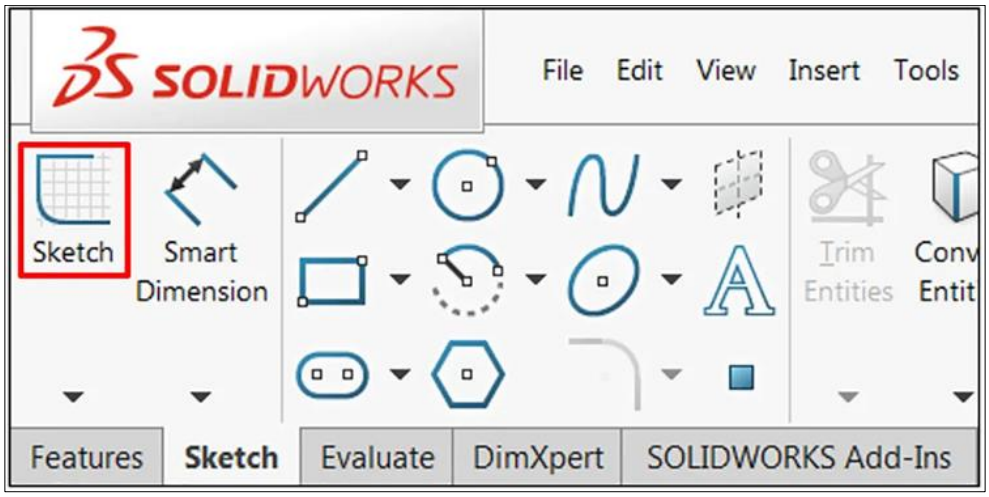


Image by Jacob Ames.

4. Screen should automatically load a triple plane diagram with “Front Plane, “Top Plane,” and “Right Plane.” **Click “Front Plane.”** The screen will automatically turn to that point of view of that plane.

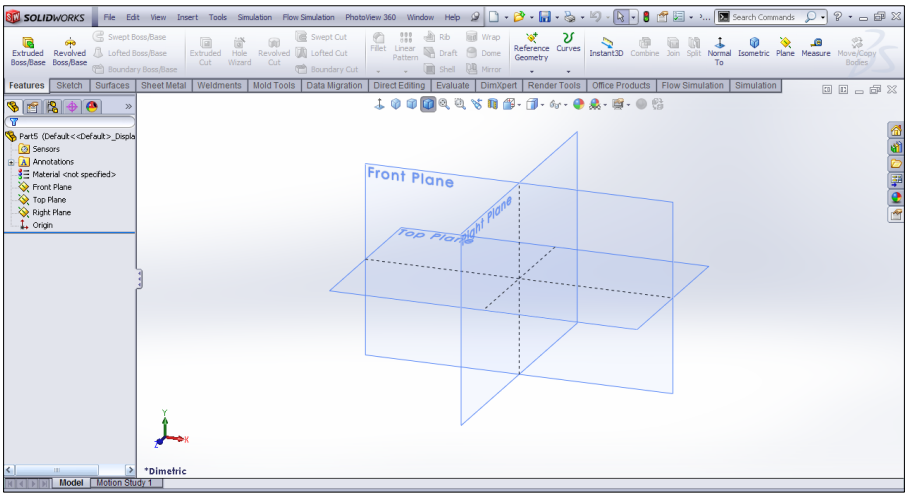
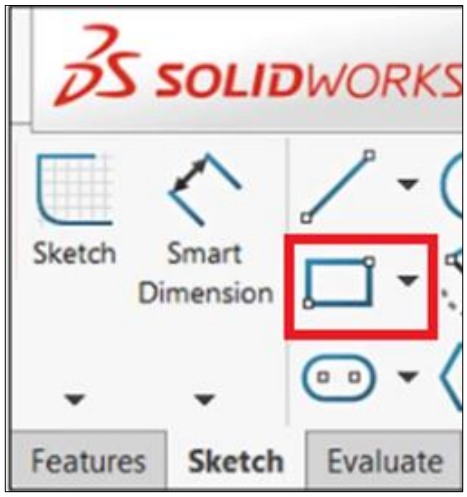


Image by GrabCAD.

5. **Click the square-rectangular icon, “Rectangle,” that appears underneath the “Line.”** It is the second item down on the first column next to the “Smart Dimension” icon.
6. **A drop-down menu will appear, click “Center Rectangle.”**



7. A small red arrow facing right (the x-axis) and one facing up (the y axis) will have already appeared in the center of the front plane. **Where the two arrows connect, the origin, click that corner.**

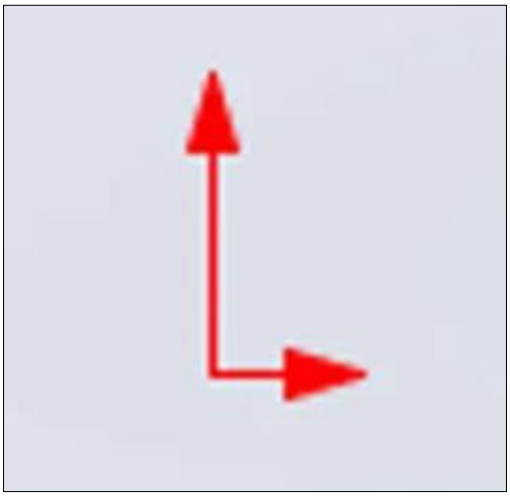


Image by Jacob Ames.

8. **Drag the curser away from the origin.** A rectangle should appear, changing in size till the mouse clicks another point on the plane.
9. **Click anywhere causing the rectangle to stop moving and appear blue on all sides.**
    a. Blue means undefined measurement, meaning that the sketch or relationship is not confirmed, accurate, or permanent. Until setting defined dimensions, elements will remain blue. When other parameters are defined in a sketch, it can cause for untouched elements to turn black (defined) by default.

i. Example: By defining two adjacent sides of a rectangle, the other two undefined sides will turn black since the opposite sides of a rectangle are the same length.

10. **Click the "Smart Dimension" icon back at the top from the toolbar, then one of the sides of the rectangle.**

11. **Move the cursor away from the selected line, causing a blue measurement to appear next to it.**
12. **Click anywhere on the plane to set the measurement in place.** A type box will appear in place of the defined measurement. **T**
13. **Type desired length of side in box then press "enter" on keyboard or click anywhere outside on the plane.**
    a. Defined side, opposite side, and Smart Dimension measurement will turn black.

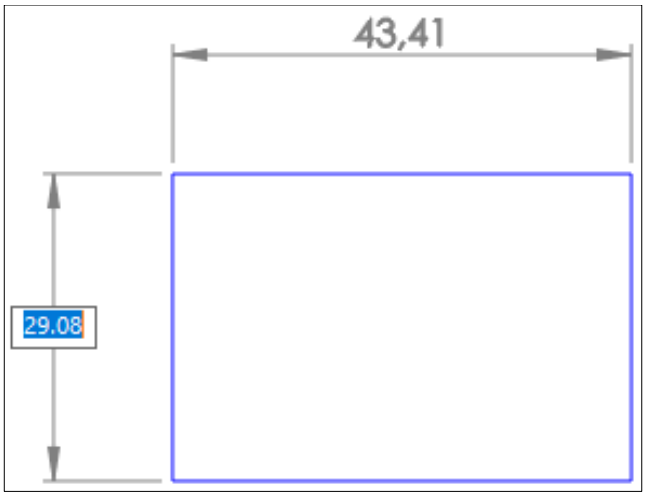
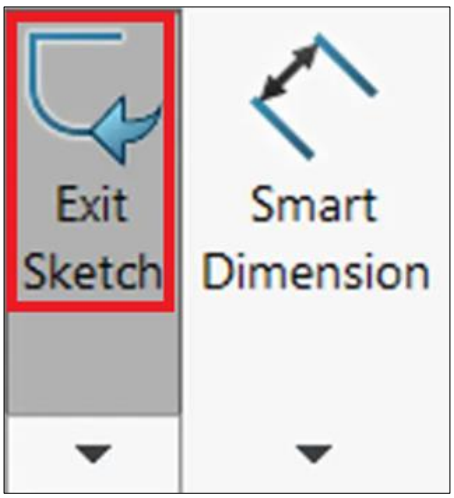


Image by The SolidWorks Blog.

14. Repeat step 8 on remaining sides of rectangle till all sides turn black.

15. In the top lefthand corner, first in the toolbar, **click "Exit Sketch."**
    a. After clicking this, the icon will return back to "Sketch" icon.



16. **Click "Features,"** the first tab in the toolbar.

Image by Perception Engineering.

17. **Click "Extrude Boss/Base."** View will automatically rotate into a yellow preview of what the sketch, the rectangle that was drawn, will look like once extruded. In the left side bar, a text box will appear to set the dimension of how much it will be extracted.
    a. It will automatically give dimensions of the preview extract.
    b. Drop down menus will automatically be set to "From: Sketch Plane" and "Direction 1: Blind." Leave these alone.

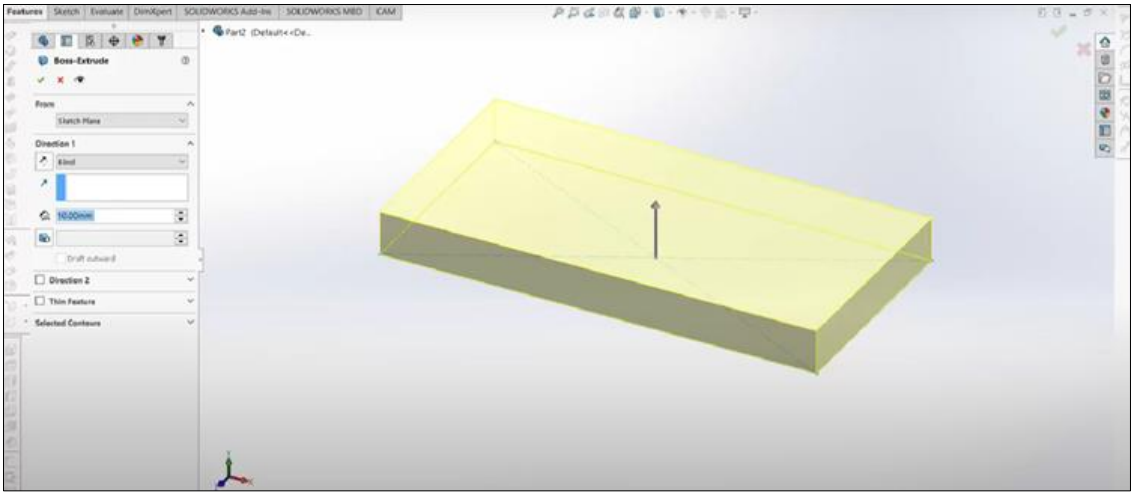


Image by Duke Electric Vehicles.

18. After typing in dimensions, click the green check mark at the top of the left side bar.

19. Depending on where the mini toolbar is located, either on the top beneath the main toolbar or on the left side of the screen, there is a rainbow sphere icon. **This is "Appearances, Scenes, and Decals," click this icon.**

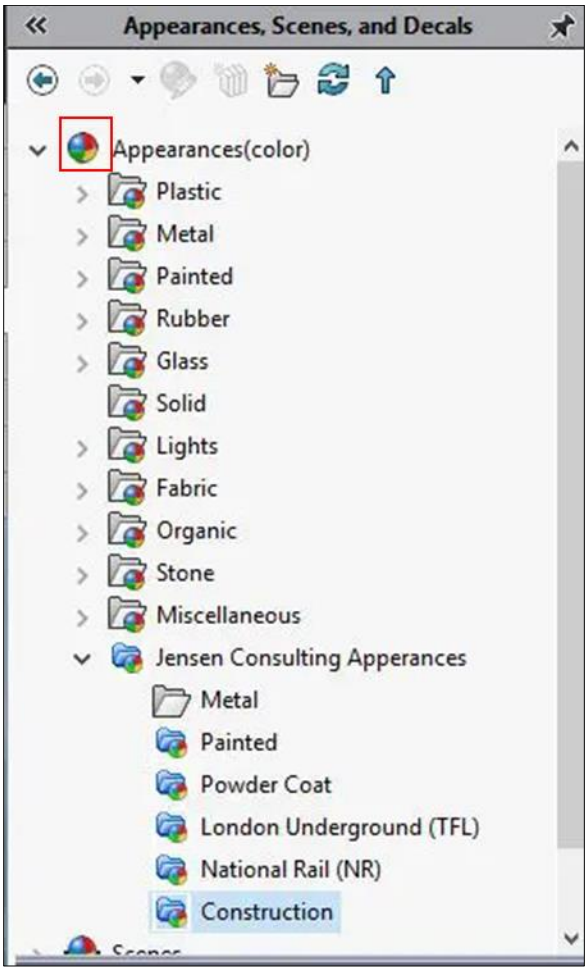


Image by Jensen Consulting.

20. Menu will open showing a list of materials. **Scroll till the folder "Wood" appears and click.**
21. **From the list of options, click the pine wood option.** Appearance will change, but dimensions will remain unaltered.

Image by McNeel Forum.

*Type of wood or material does not affect drawing but may affect catapult as a result of different thickness and density. Applying the wood material in SOLIDWORKS is only a visual element for aesthetics and not for 3D output testing. SOLIDWORKS is only a modeling tool that can preview and design whether physical elements are compatible for 3D use.

22. After being left with a defined wooden block, **save**.
    - **a.** Save, save, save. Save any and all work throughout the design process in SOLIDWORKS. SOLIDWORKS does have an automatic save option that will automatically save every few minutes, but there is no guarantee that it will save any life-threatening changes that have been made. Always save work after making important changes.
23. **Click the "Save" icon that is on top of the toolbar. Name the file "Wood Block" and save part to desktop or flash drive.**
    - **a.** File can be saved through the keyboard option too (CTRL + S).

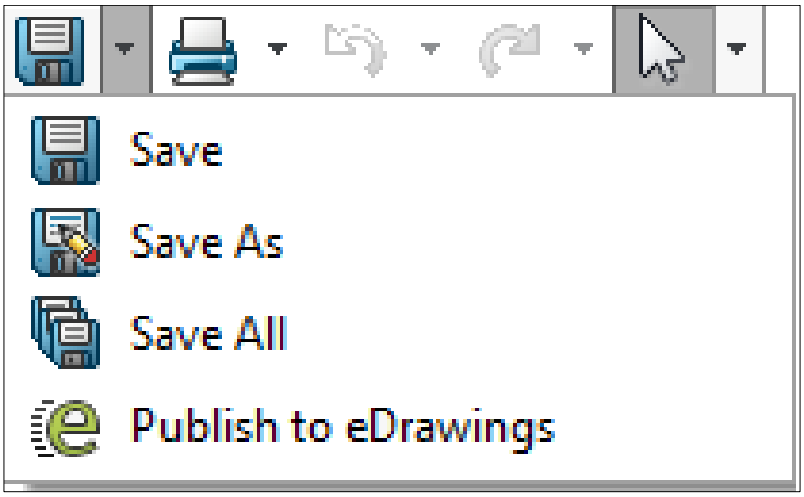


Image by SolidWorks.

Assembling the Blocks Together

1. After part saves, **go back into SOLIDWORKS click "Assembly" and then "OK."**

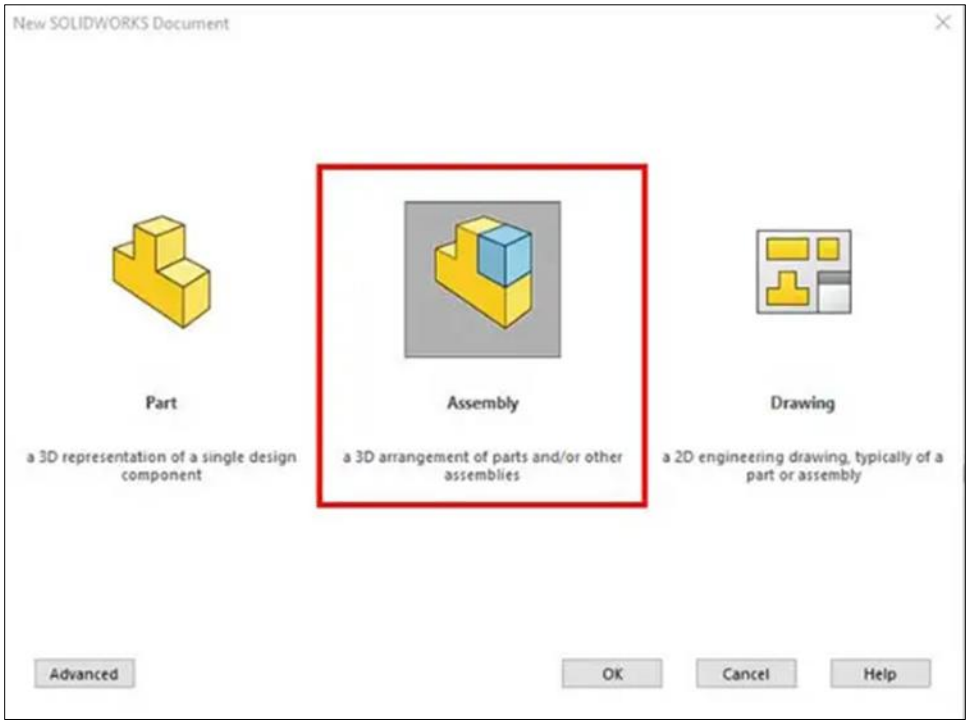
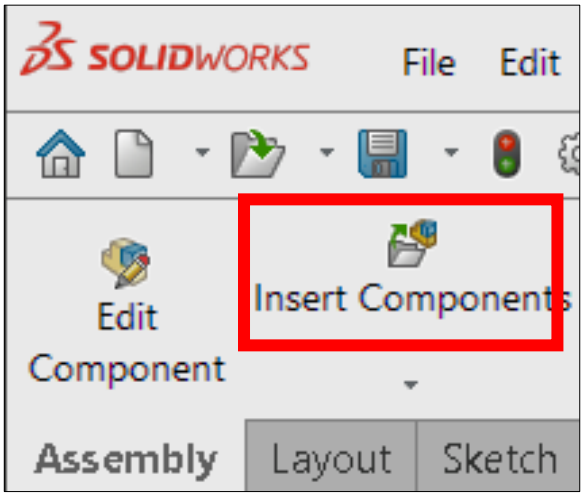


Image by Hawk Ridge Systems.

2. On the left side bar, **click "Browse."**
3. This will open up file explorer, **click the saved part "Wood Block" and then "Open."**
4. Part will loosely appear in empty plane of screen, **click anywhere on plane to place it.**
    a. This can be moved later, but for now it will not affect the assembly.
5. In the top lefthand corner of the top toolbar, **click "Insert Components" icon.**



6. **Repeat step 3 once.**
7. Next to "Insert Components" icon, **click the paperclip icon "Mate."**

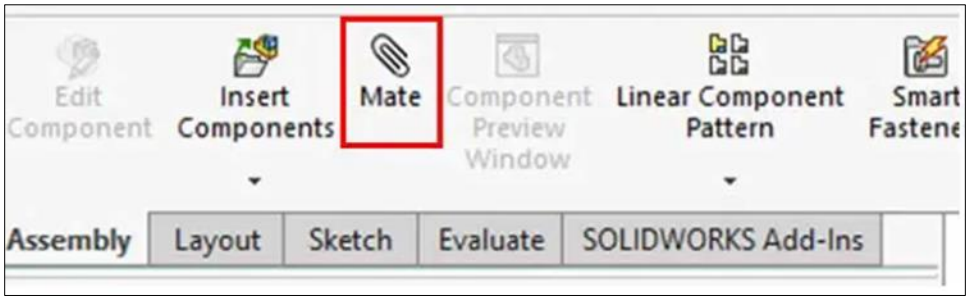
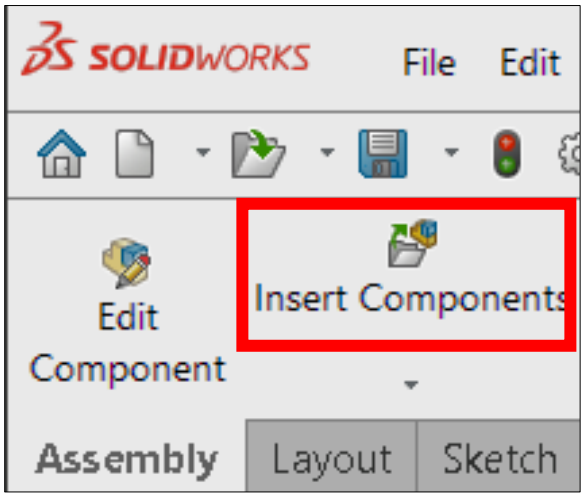


Image by Hawk Ridge Systems.

8. **On wood block 1, click the face of one of the shorter sides**, this will highlight that face blue.
9. **Click the face of one of the long sides of wood block 2**, highlighting it in orange.
10. **Click the green check mark that appears at the top of the left sidebar.**
    a. This will bring these two faces together, but they will still be able to move freely apart from each other. This mate puts the two faces together like two cars on the same road. They are now facing and on the same level but can still move whichever way they want.
11. **Click "Mate" and then click the one side from each wood block where the wood pieces will align.**
12. **Click the green check mark that appears at the top of the left sidebar.**
    a. This will centralize the movement of the wood blocks to move only up and down in reference to the shared face and side.
13. To make the block immovable and secure, **click "Mate."**
14. **Click a side from both wood blocks where the wood pieces will align.** Make these sides adjacent to the previous Mate that was made in step 4.
15. **After selecting the adjacent sides, click the green check mark at the top of the left sidebar.** This will lead to two sides and one face coming together causing the two wood blocks to be connected and immovable.
    a. In some cases, mates can also be undefined (blue) and then defined (black).
16. **Save the part with the two wood blocks so far. Name the assembly file "Square Base" and then save it to desktop or a flash drive.**
    a. There is an option to save assembly and continue working on it without it closing the assembly file altogether. If it remains open, move on to step 10.
    b. If it closes, click on the assembly file from wherever it was saved and reenter it. Since this is already a saved assembly, the menu screen with the three options will no longer appear.

17. * There is the option of saving after every individual mate is complete. Some engineers find this helpful for sake of peace of mind and perform this efficiently with the keyboard shortcut. This is only preference, but always a safe option.
18. **Click "Insert Components" and insert "Wood Block" once.**



   a. Two wood blocks can be added but are not recommended for beginners. To make it easier to keep track of all Mate relationships with specific pieces one part at a time is best, especially when all the parts look the same.

19. **Repeat steps 1 - 10.** This will result in three wooden blocks to be connected in the shape of a "C."
   a. Save assembly file.

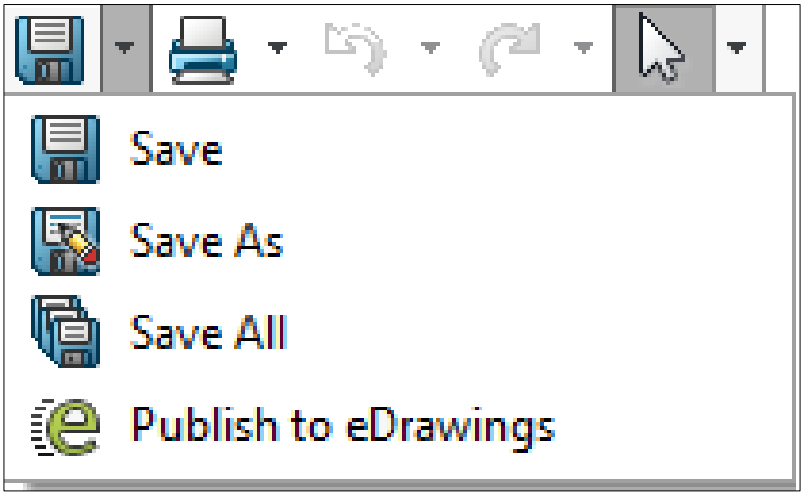
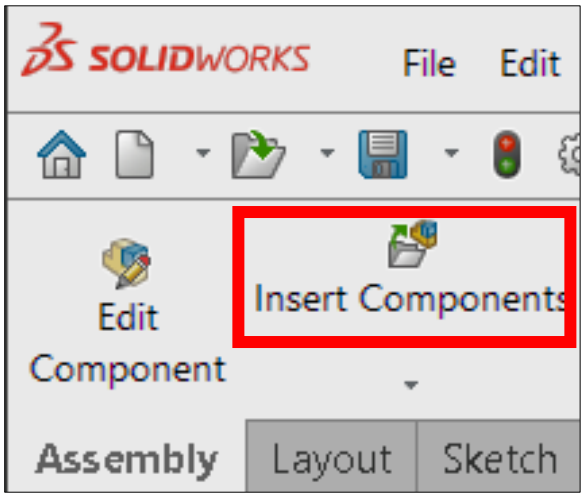


Image by SolidWorks.

20. **Click "Insert Components" and insert "Wood Block" once.** This will be the final piece to close the base in a perfect square shape.



21. **Repeat steps 1 - 10**. This will result in the completed square base made out of wood for catapult.
22. **Save finished assembly file.**

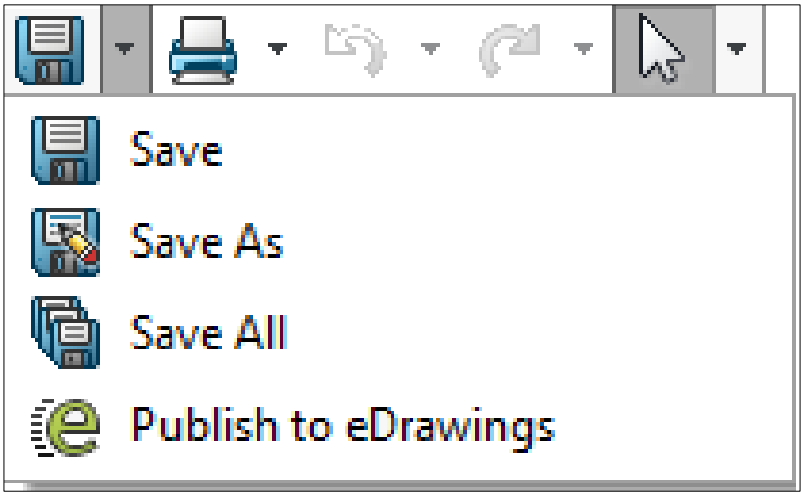


Image by SolidWorks.

* For the rest of catapult drawing, create a separate assembly file. This way with previously made parts already defined and measured, it makes the mating process quicker than drawing and extruding directly on an assembly.

Creating a Rectangular Catapult Base

Depending on the type of catapult being built or designed, the base shape can be different. This next set of instructions will accommodate for a rectangular base shape. Software will be the same instructions for SOLIDWORKS 2019 but can be used for other SOLIDWORKS software versions.

For more information on downloading SOLIDWORKS or updating to the latest version, please visit https://www.solidworks.com.

1. **Repeat the steps from "Making the Parts" except for the naming.**
2. **Name part file "Large Wood Block."**
3. **Open SOLIDWORKS and create a new part.**

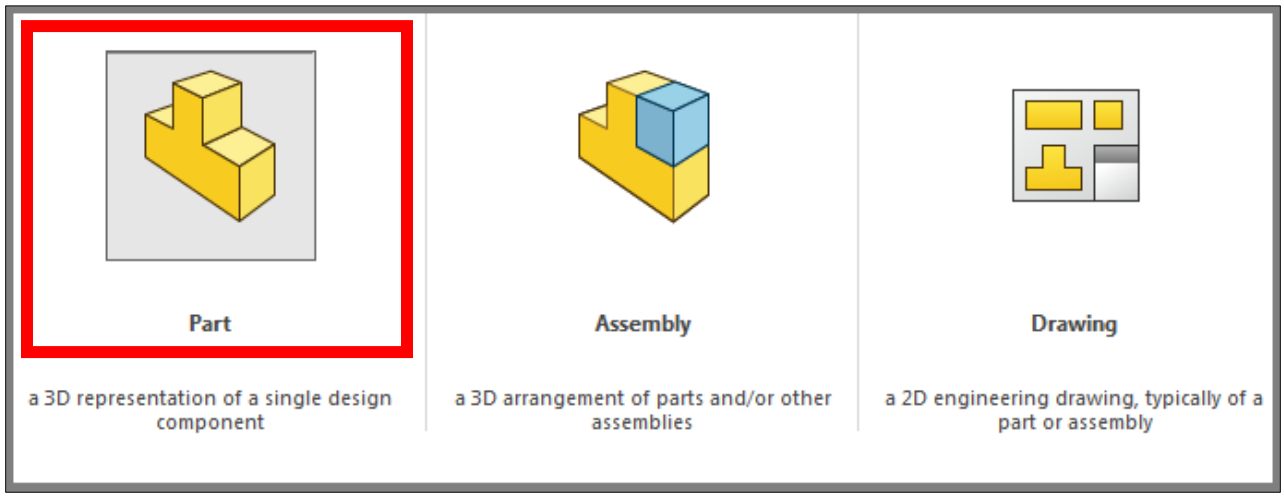
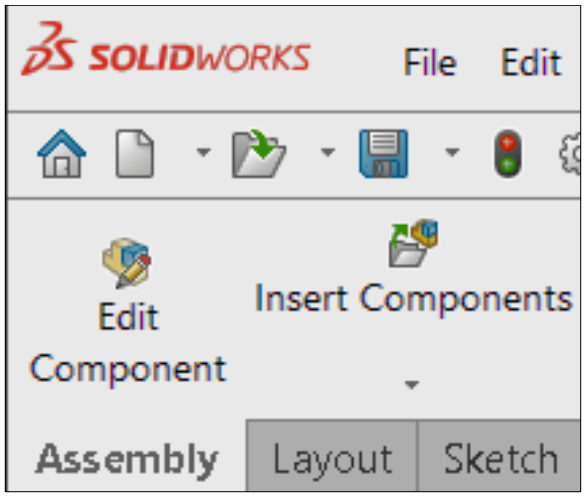


Image by Angelle Erickson.

Repeat step 1 but make this new part a smaller rectangle in length. This will allow for the catapult base shape.

4. **Name part file "Small Wood Block."**
5. **Repeat steps 1 – 10 from "Assembling the Blocks Together", but instead of inserting two wooden blocks, insert one "Large Wooden Block" and one "Small Wooden Block."**



6. **Save new file.**

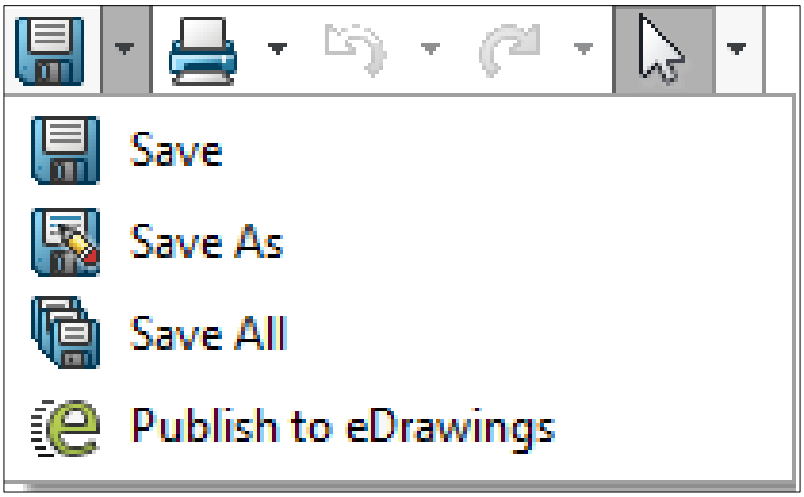


Image by SolidWorks.

* Product tests for catapults with both bases are recommended since the top half of catapult is directly affected by the base. This could result in a change in performance: projectile launching, projectile targeting, release strength, release distance.

Cutting Out a Circular Hole in Wood (For a Pipe)

Every catapult needs to have some sort of cylindrical object, like a pipe, that will allow the arm of the catapult to swing up. In order to accomplish this, circular holes are cut into two sides of the wood and one side of the arm of catapult allowing a pipe to fit through. The following directions will be with the assumption that pipe hole will be cut into the longer wood blocks.

* Pipes are the most common resource to catapult building since they are cheap and easy to find, allowing catapult building to be an easier less stressful process.

Directions

1. **Click open assembly file named "Square Base."**
2. **Rotate assembly so the view is facing one side.**
3. **Click on one of the sides with a face that's on the outside of one rectangle**, highlighting that face.
4. **Click "Sketch" in toolbar.**

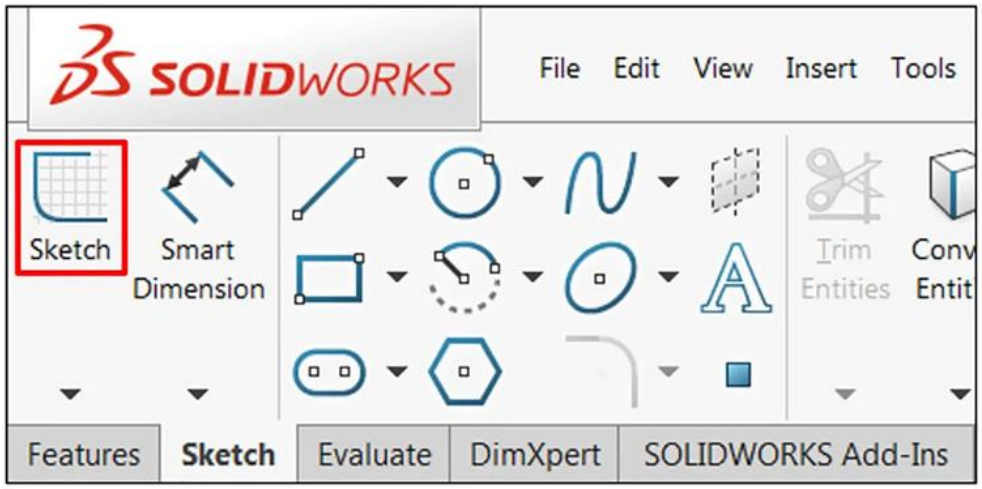


Image by Jacob Ames.

   a. Dropdown menu "From" should automatically be on "Sketch Plane." This puts the drawing directly on the assembly.

5. **Click the "Circle" icon.**

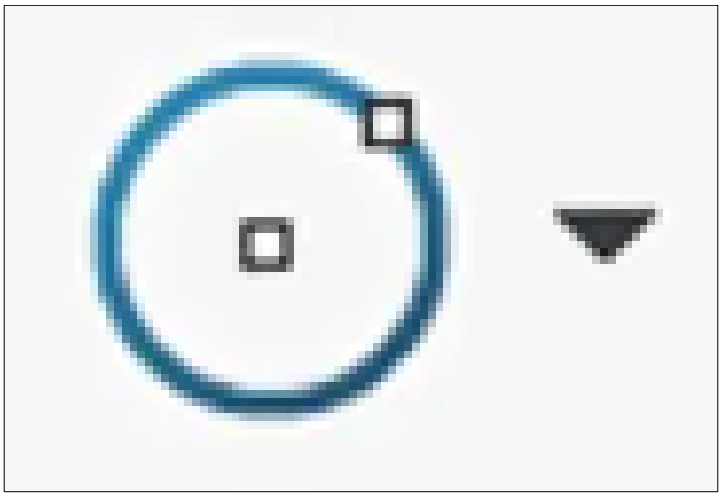


Image by Hawk Ridge Systems.

6. **Click anywhere on the face where sketch is being drawn on.** This will create the center point of circle.
7. **Drag cursor out creating the outline of circle and click anywhere.**
    a. Circle will be blue.
8. **Click "Smart Dimension" then click on circle outline.**
9. **Drag cursor out as dimension line appears.**
10. **Click anywhere, the type box will appear.**
11. **Type the diameter of the circle and then click the green check mark.**

    a. Only the size of the circle is defined now.
12. **Click "Smart Dimension," the circle outline, and then a side that is next to the circle on the plane.** This will specify how far the circle's outline is from that side.
13. **Type in distance**

14. Click the green check mark.

a. The dimension itself will appear black, but the circle will still be blue. This is because the circle needs two defined dimensions from adjacent sides specifying where it is on the assembly's plane.

15. Click "Smart Dimension," the circle outline, and then an adjacent side to the one that was selected previously in step 8.

16. Type in distance the circle outline from that side

17. Click the green check mark.

a. Circle should now appear black.

18. Click the "Features" tab in toolbar.

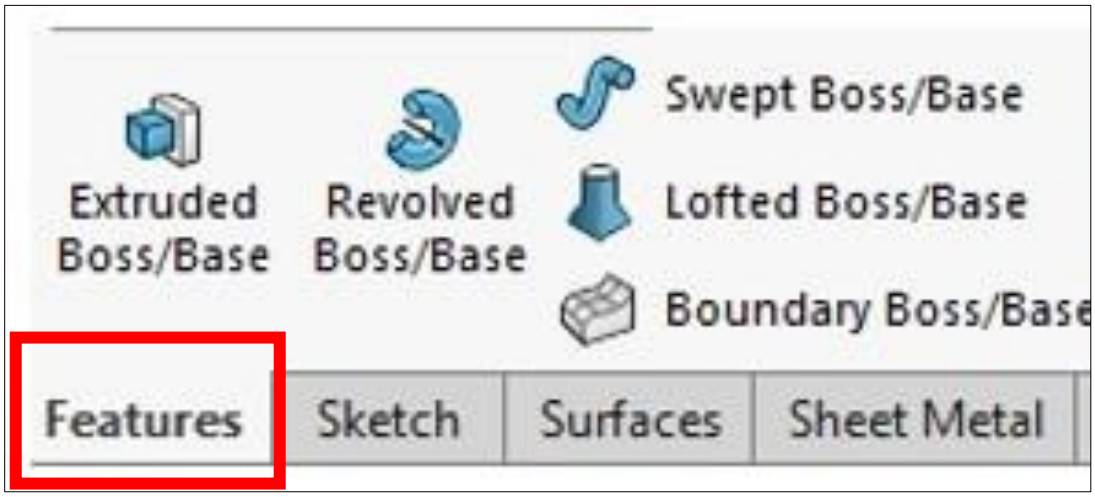


Image by Perception Engineering.

19. Click on the circle outline.

20. **Click the "Extrude Cut" icon in toolbar.** The Assembly will show a yellow preview of the shape and size of the cut as well as the direction it will go in.

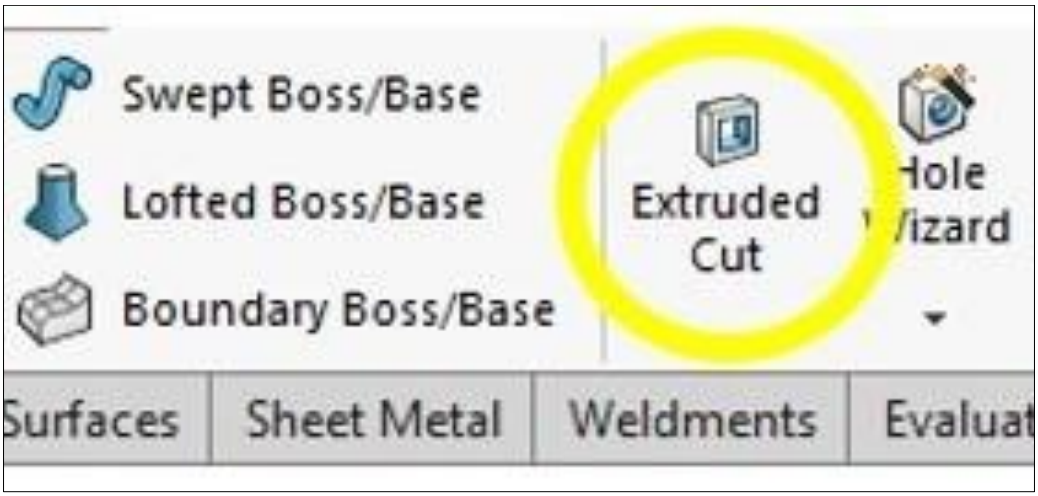


Image by Perception Engineering.

21. **Click the grey arrow that appears on top of the yellow preview and extend it**, so it cuts all the way through the base on both sides of the two opposite wooden blocks.
22. **Click the green arrow.**
23. **Save assembly with new cut.**

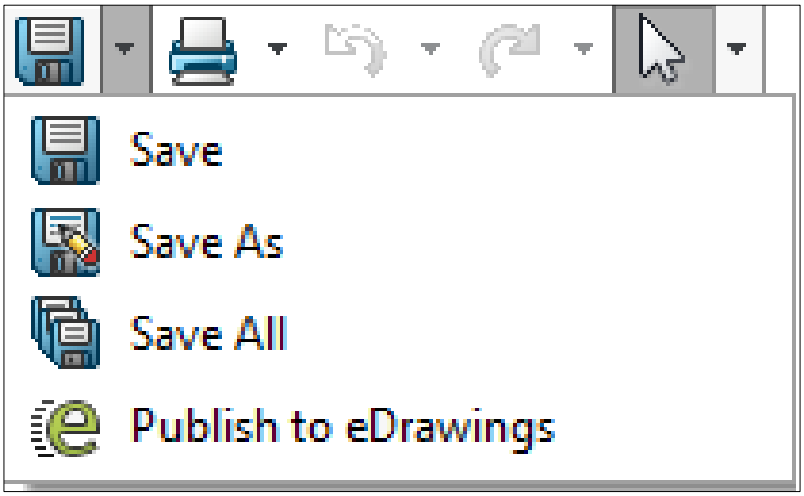
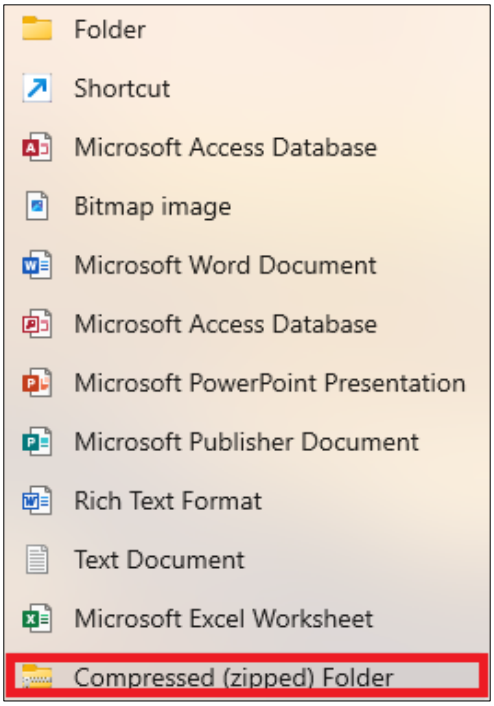


Image by SolidWorks.

Zip Files

Zip files are an important aspect of sharing files, especially when you are working with other people. You will find that after adding so many elements to your parts and assemblies that the files take up a lot of space. Instead of emailing them as is, putting all your files into a compressed zip folder will allow you to be able to send more files at once. It will also take up less space. Receivers can easily download the zip folder in less time and won't lose anything in the process.



Conclusion

With the evolution of technology and what it means to design, this tutorial aims to help future engineers build a better tomorrow. Through exploring how to create and design different objects, understanding the importance of 2D design in reference to 3D design, and by learning new tools that can help make your projects more accurate in engineering, following this tutorial can help you continue your SolidWorks journey.

Check https://www.solidworks.com/ for more designing ideas.

FAQs

For any questions or concerns check out https://www.solidworks.com/support/frequently-asked-questions.

Trouble Shooting Guide

Please follow the directions linked below by SolidWorks edition that is downloaded.

2021:
https://help.solidworks.com/2021/English/SolidWorks/sldworks/c_SOLIDWORKS_Troubleshooting_Overview.htm

2022:

https://help.solidworks.com/2022/english/SolidWorks/sldworks/c_solidworks_troubleshooting_overview.htm

2023:

https://help.solidworks.com/2023/english/SolidWorks/install_guide/c_troubleshooting_upgrades.htm

Credits

"AutoCAD Architecture Toolset: Architectural Design Software." *Autodesk*, www.autodesk.com/ca-en/products/autocad-architecture?plc=ACDIST&term=1-YEAR&support=ADVANCED&quantity=1. Accessed 12 Nov. 2023.

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Mufasu CAD. "Solidworks How To Change Material." YouTube, 1 March, 2023, https://www.youtube.com/watch?v=MDrtPGZKpL4&ab_channel=MufasuCAD.

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Solidworks Master. "how to draw circle in solidworks and use extruded boss, extended cut." YouTube, 1 March, 2023, https://www.youtube.com/watch?v=lhwrPRYoTnA&ab_channel=SolidworksMaster.